J. Inst. Eng. India Ser. C
DOI 10.1007/s40032-017-0402-x

ORIGINAL CONTRIBUTION

Performance Evaluation of Nose Cap to Silica Tile Joint of RLV-TD under the Simulated Flight Environment using Plasma Wind Tunnel Facility

Aravindakshan Pillai[1] · K. Krishnaraj[1] · N. Sreenivas[1] · Praveen Nair[1]

Received: 16 May 2017 / Accepted: 9 October 2017

Abstract Indian Space Research Organisation, India has successfully flight tested the reusable launch vehicle through launching of a demonstration flight known as RLV-TD HEX mission. This mission has given a platform for exposing the thermal protection system to the real hypersonic flight thermal conditions and thereby validated the design. In this vehicle, the nose cap region is thermally protected by carbon–carbon followed by silica tiles with a gap in between them for thermal expansion. The gap is filled with silica fibre. Base material on which the C–C is placed is made of molybdenum. Silica tile with strain isolation pad is bonded to aluminium structure. These interfaces with a variety of materials are characterised with different coefficients of thermal expansion joined together. In order to evaluate and qualify this joint, model tests were carried out in Plasma Wind Tunnel facility under the simultaneous simulation of heat flux and shear levels as expected in flight. The thermal and flow parameters around the model are determined and made available for the thermal analysis using in-house CFD code. Two tests were carried out. The measured temperatures at different locations were benign in both these tests and the SiC coating on C–C and the interface were also intact. These tests essentially qualified the joint interface between C–C and molybdenum bracket and C–C to silica tile interface of RLV-TD.

Keywords RLV-TD · Carbon–carbon · Silica tile · PWT · Thermal protection system

✉ Aravindakshan Pillai
aravind_vssc@yahoo.co.in

1 Vikram Sarabhai Space Centre, Indian Space Research Organisation, Trivandrum, Kerala, India

Introduction

ISRO has successfully flight tested the reusable launch vehicle through launching of a demonstration flight known as RLV-TD HEX mission. One of the main objectives of this mission is to evaluate the performance of the thermal protection system materials under the re-entry thermal environment. In the RLV-TD, depending on the thermal environment prevailing at various regions during flight, different types of thermal protection materials have been used. All the leading edges of the RLV-TD are made of 15CDV6 except the vertical tail leading edge which is made of Inconel. The Elevon, which is a control surface is made of Titanium, the leeward side is thermally protected by flexible external insulation (FEI). The nose cap region (which is the main focus of this work) is thermally protected consists of carbon–carbon as the hot structure followed by silica tiles with a gap in between them. The gap between the C–C nose cap and silica tile is filled with silica cloth. C–C nose cap is interfaced with nose body using four metallic molybdenum clamps, which interface with aluminium fore end ring of the nose body (Fig. 1). Silica tile with strain isolation pad (SIP) is bonded with high temperature ceramic adhesive on aluminium FE ring. The gap between C–C nose cap and silica Tile is filled with silica fabric and kept in position by bonding using high temperature ceramic adhesive. To evaluate and qualify this interface under the simultaneous heat flux and shear flow conditions, tests were carried out in plasma wind tunnel (PWT) facility wherein the entire joint between the C–C and silica tile including the metallic interfaces were simulated. Before carrying out the actual model tests, facility calibration was carried out to arrive at the operating conditions to simulate the required heat flux and shear flow conditions. A double wedge model was chosen to simulate

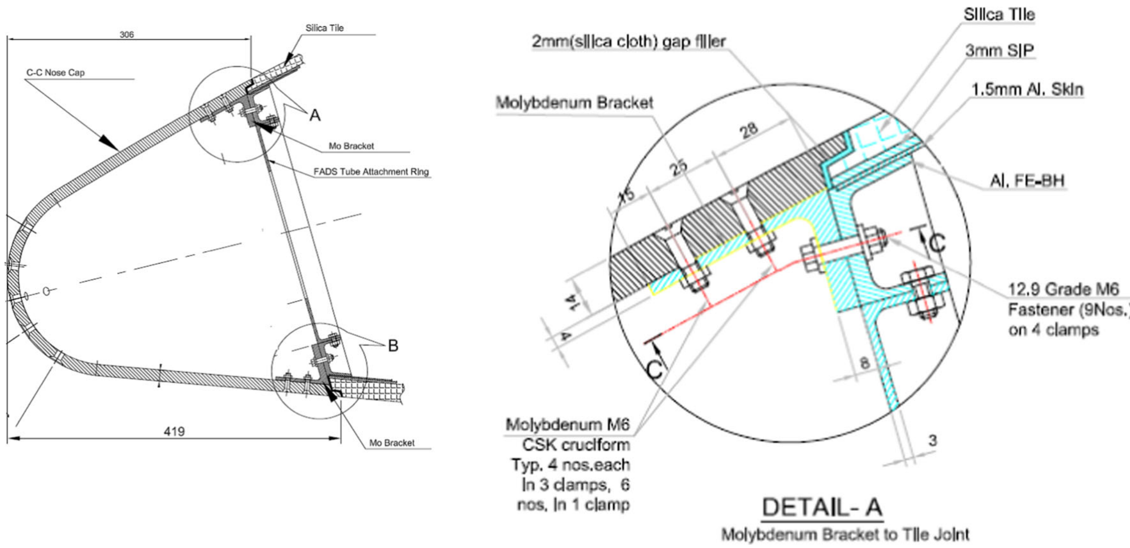

Fig. 1 C–C nose cap interface configuration

the heat flux and shear flow conditions. Here the entire interface was simulated as in the flight. Two tests were carried out with this model. In the first test, the test duration was limited to 22 s to simulate the total heat load expected in the flight. In the second test, the test was carried out for a duration which corresponds to the front wall surface temperature of C–C reaching 180 °C. It may be noted that the maximum temperature limit is 3000 °C for C–C, 450 °C for molybdenum and 1400 °C for silica tile. The measured temperatures during the test at different locations were benign and the SiC coating on C–C and the interface were also intact.

Testing Requirements

The main objective of the test is to simulate the Interface joint between C–C, silica tile and molybdenum bracket of the RLV-TD nose cap region and evaluate the final temperatures of the same under the simulated flight conditions. The interface consists of the carbon–carbon which is dry bonded with molybdenum bracket using four nos. of M6 screws made of molybdenum. A tightening torque of 3 N-m is given to the bolts. The silica tile is bonded to strain isolation pad which in turn is bonded to the aluminium base plate. There is a gap of 2 mm between the C–C to silica tile joint which is filled with gap filler. During the ascent and re-entry flight, the heat flux prevailing at C–C to silica tile joint is shown in Fig. 2. As can be seen the maximum heat flux at the interface is less than 5 W/cm^2 and the total heat load is 197 J/cm^2. The peak shear stress predicted is 120 Pa. Generally this type of system level qualification is carried out in plasma wind tunnel facilities under the simulated reentry conditions

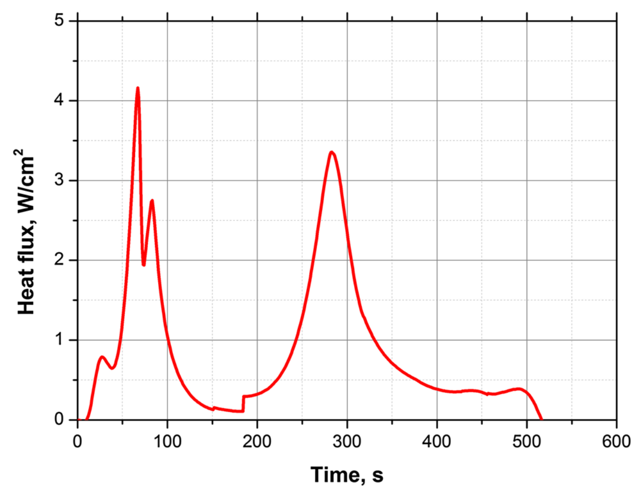

Fig. 2 Heat flux history in the C–C to silica tile interface

[1–3]. The 6 MW plasma wind tunnel established in Vikram Sarabahi Space Centre (VSSC), ISRO is made use of for the testing of RLV-TD nose cap to silica tile interface [4, 5].

Facility Description

PWT facility consists of a 6 MW rated segmented constrictor plasma generator with a hypersonic nozzle and an environmental simulation system for duplicating the high temperature, high enthalpy plasma flow. Facility subsystems are designed to operate for a wide range of conditions so that all re-entry conditions can be simulated in the facility. The photograph of the PWT facility is shown in Fig. 3. Major subsystems of the facility are plasma generator, hypersonic nozzle, power supply system, coolant supply system, gas feed

Fig. 3 Photograph of the PWT facility

system, test chamber, and water cooled supersonic diffuser duct, heat exchangers, model injection system and vacuum pumping system. Blunt body tests simulate stagnation point heat transfer and blunted wedge tests simulate surface heat transfer and shear flows [1–3]. A wide range of simulation capabilities are achieved by carefully choosing the model configuration, nozzle area ratio and the plasma generator operating conditions. Test models of different shape and configuration can be tested in the facility, but overall size is decided by tunnel blockage and the simulation requirements. Summary of overall simulation capabilities of the facility is given in Table 1.

Test Scheme

In PWT facility, the high temperature ionized gas is produced by heating air using high power electric arc maintained between the electrodes of the plasma generator. This ionized species, namely, plasma is allowed to expand in a CD nozzle which is open to a test chamber, maintained at high vacuum condition enables plasma to come out as a high velocity jet. The test model is positioned in front of the nozzle exit at appropriate axial location so that the high velocity plasma flows over the model and simulates the re-entry conditions. During testing, the test chamber is maintained at high vacuum condition by making use of hypersonic diffuser duct and vacuum spheres together with pumping system. The model configuration and the operating conditions are finalized using facility calibration.

Table 1 Simulation capabilities

Stagnation temperature	6000–9000 K
Stagnation pressure	10–200 m-bar
Stagnation enthalpy	20–40 MJ/kg
Steady state heat flux	10–300 W/cm^2
Model size	Φ125 mm dia stagnation model; shear model 200 mm × 300 mm × 50 mm
Duration	Up to 1000 s

Facility Calibration

Prior to test, it is required to arrive at facility operating conditions necessary to achieve desired heat flux and shear levels on test article. This is done by mapping the heat flux on models that are geometrically similar to the actual test article. Since the simultaneous simulation of shear flow and heat flux is required, a double wedge calibration model was chosen. Calibration tests using shear models with different wedge angles were carried out and found that model with 9° and 7° half angle can simulate the required heating level. The operating condition and the test model were numerically simulated and the possible shear stress during testing is computed and is in line with the requirements. The actual test model will be identical to the calibration model in all respects. The details of the model are shown in Fig. 4. Carbon phenolic insert is used as the leading edge and rest of the body is made out of medium density ablative (MDA). Six numbers of heat flux gauges are provided in the 9° surface and four numbers of heat flux gauges are provided on the 7° surface. The calibration model is located in front of the nozzle with respect to the axis of the nozzle. It is then moved away from the nozzle centerline, using a remotely controlled actuator. The test chamber and diffuser duct are evacuated to 0.1 m-bar. The plasma generator is started and the power is increased gradually to required level. The gas flow is increased gradually. Once stable operating conditions are reached at the preset power condition, the model is injected into the center of the plasma jet. It will be maintained there for the required duration and the thermal response will be continuously monitored/recorded through embedded thermocouples. For the present case, the calibration was carried out by injecting model into plasma jet when the facility was operating at 1150 A level. The model was located at a distance of 385 mm from the nozzle exit. At this operating condition, air is heated to a total temperature of 7850 K. The model is kept in the plasma jet for 6–7 s. The temperature rise of the

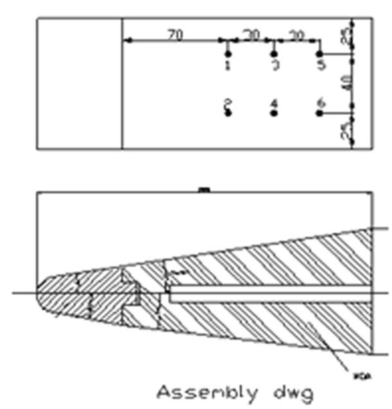

Fig. 4 Calibration model

Fig. 5 Calibration model in the test chamber

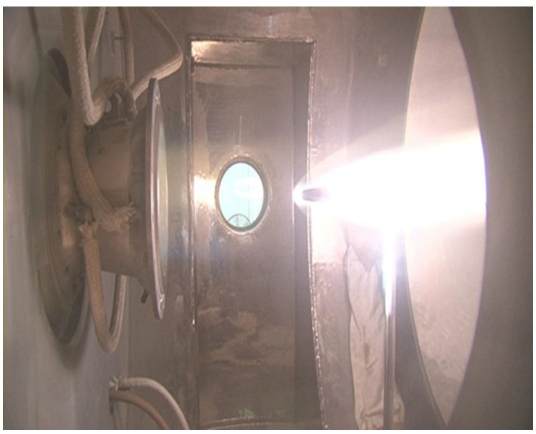

Fig. 6 Testing of the calibration model

copper slug is measured during its exposure to plasma. The heat flux at the surface is computed from the temperature gradient and it varied between 7 and 9 W/cm^2. The photograph of the calibration model and the testing of the calibration model are shown in Figs. 5 and 6.

Model Testing Methodology

Based on the calibration results, the test model is realized with the same configuration as that of the calibration model. The leading edge of the model is positioned exactly at a distance of 385 mm from the nozzle exit as in the calibration test. All thermocouples are connected to the data acquisition system and checked for its working. The test is started with arc initiation through a high frequency unit, followed by gradual increase of power and flow rates. Once the plasma is operating at 1150 A the model will be injected into the plasma stream and simulates the above heating conditions. The model will be kept in the jet for a duration corresponding to the total heat load in the flight. Thermal behaviour of the joint will be understood from the measured temperature of the model and the post test inspection.

Details of the Test Model and Instrumentation

The test model is realized with the same configuration as that of the calibration model. The leading edge is made up of carbon phenolic and the 9° surface of the model is used for simulating the C–C to silica tile interface together with molybdenum brackets. The details of the model are shown in Fig. 7. The SiC coated C–C piece is connected to the molybdenum bracket using four molybdenum screws and the applied torque is 3 N m. The silica tile was bonded to the SIP and is fixed to aluminium bracket. The Z-shaped interface between the C–C and the silica tile is filled with gap filler as in flight. In order to evaluate the thermal performance of the joint interface, 10 numbers of K-type thermocouples were provided to measure the C–C back wall temperature, temperature of the molybdenum bolts, interfaces and the brackets. The locations of the thermocouples are also shown in the Fig. 7. Table 2 explains the exact location of thermocouple.

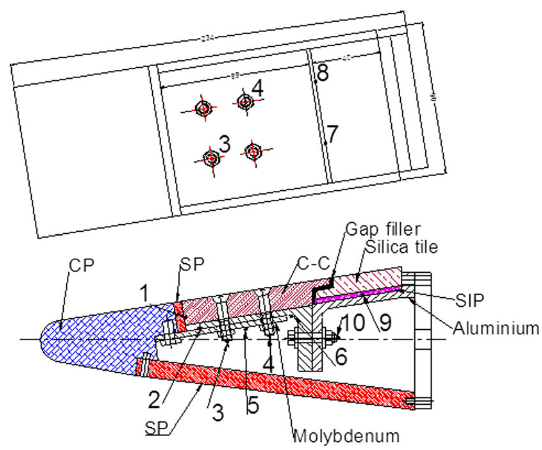

Fig. 7 Model with instrumentation details

Table 2 Thermocouple locations

Thermocouple ID no.	Thermocouple location
1	Carbon–carbon
2	Molybdenum bracket
3, 4	Molybdenum bolt
5	Molybdenum base plate
6	Molybdenum bracket
7,8	Silica tile-gap interface
9	SIP-aluminium interface
10	Molybdenum bolt

Determination of Heat Flux and Shear Stress on the Model

Numerical analysis was carried out to determine thermal and flow parameters on the model during testing. The simulation of flow field around the test model and there by the determination of the heat flux and shear stress is carried out using unstructured finite volume based Reynolds averaged Navier-Stokes solver. Even though the flow inside the nozzle and the flow over the model are laminar, one-equation Goldberg turbulence model is also solved along with other NS equation for the complete understanding of the flow physics. Simulations are carried out in two phases. In the first phase, simulation is carried out for plasma facility without model. Entire computational domain is initialized with 1 Pa and 300 K with a very low velocity of around 0.001 m/s. Inlet pressure and temperature are gradually increased to chamber conditions (3.26 bars and 7900 K). Density error has come down by around three orders and hence the flow parameters presented below are converged solutions.

Figure 8 shows the Mach number distribution inside the computational domain. Flow expands to a Mach number of around 5 at the nozzle exit. Since the nozzle exit pressure is more than the test chamber pressure, the flow expands further in the test chamber to a Mach number of around 8.9. Expanded flow is confined to the diffuser with the help of catch cone where it undergoes a series of shocks. Mach number at the point (385 mm from nozzle exit) where the model is placed is around 5.6581 and is shown in Fig. 9 which gives the Mach number variation along the nozzle axis.

The Mach number, static pressure and static temperature along the axis of the facility in front of the nozzle is obtained and its values at the point (385 mm from the nozzle exit), where model leading edge is coming is taken as the input for 2D simulation of flow over the wedge model.

Flow and fluid properties for 2-D flow simulation over RLV-TD model are calculated from tunnel flow simulation and are presented below,

Static pressure, Pa	Static temperature, K	Mach number
85	1690	5.6581

Entire computational domain is initialized with Mach number of 5.6581, static temperature of 1690 K and static pressure of 85 Pa. These values are also given as supersonic inflow boundary conditions. Wall of the model is given isothermal no-slip boundary condition to get cold wall heat flux which will be the maximum heat flux experience by the model during the test. Figure 10 gives the Mach number plot and its contour. A bow shock formed ahead of the model is well captured without any carbuncles as seen in the Mach contours.

Figure 11 gives the cold wall heat flux over the model surface. Stagnation point heat flux is around 260 W/cm^2 and it decreases to around 10 W/cm^2 near the nose joint. The shear stress computed from the flow parameters at the joint is 78 Pa.

Theoretical Analysis

The heat flux distribution around the model predicted by CFD is used as input for 3D non-linear thermal analysis of the test model to predict the temperature at various locations. Domain considered for the analysis is shown in Fig. 12. Finite element model was generated using hyper mesh. The temperature contour plot of the full domain at the end of 22 s is shown in Fig. 13.

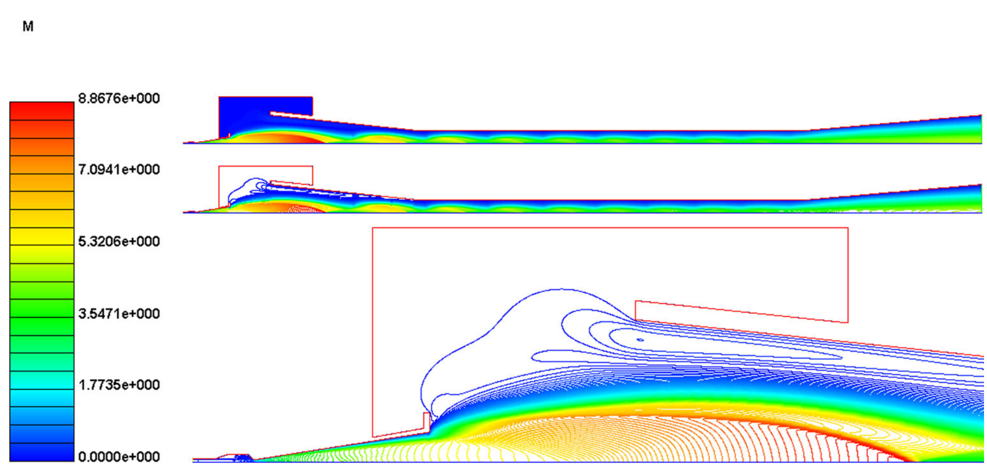

Fig. 8 Mach contours inside plasma facility without model

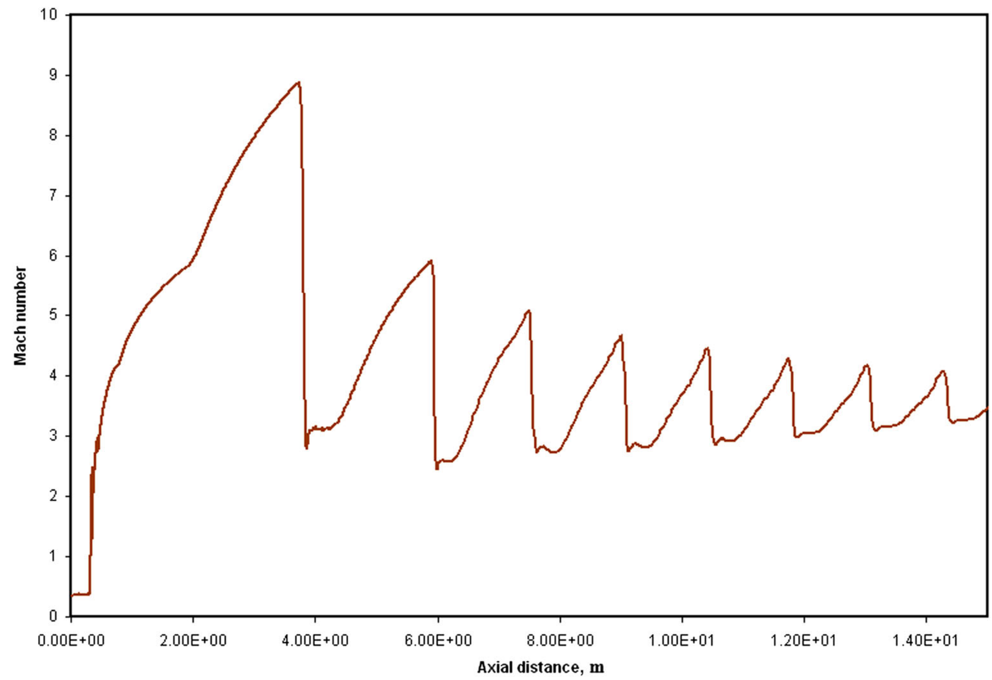

Fig. 9 Variation of Mach number along the nozzle axis

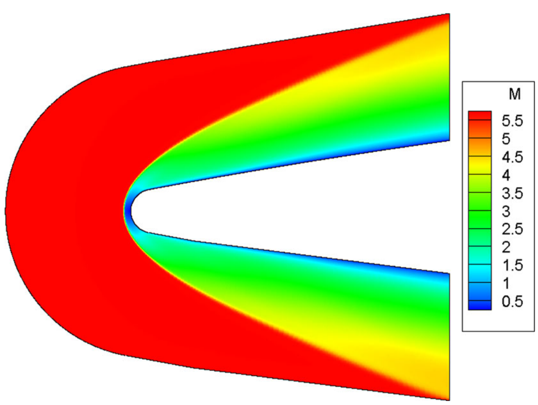

Fig. 10 Mach number plot

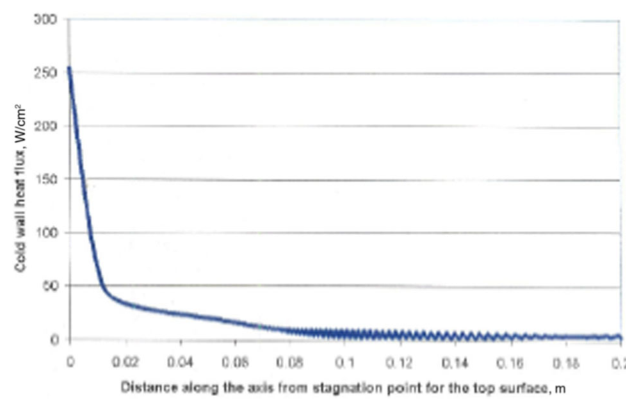

Fig. 11 Variation of cold wall heat flux over the top surface of the model

Results of the First Test

Test was conducted by injecting the model to the plasma stream when the facility was operating at calibrated operating condition. The model was exposed to 22 s to simulate the total heat load expected in flight. The photograph of the test article assembled inside the chamber and a photograph during testing is shown in Figs. 14 and 15. The temperature sensors monitor the thermal response of the test article during testing. The measured temperatures are shown in Fig. 16.

The temperatures measured in all the interfaces are quite benign. A visual inspection of the test article was carried out and found that the article was intact. No damage or erosion of the article was seen. An ultrasonic testing indicated no cracks or delamination on the SiC coated C–C specimen. No torque relaxation was observed on the molybdenum bolts after the test.

Comparison with Prediction

Table 3 shows the comparison between the predicted and measured temperatures at the end of the test.

As can be seen, there are some deviations in measured and predicted temperatures. The difference may be due to improper contact between the joints or variations from the assumed properties of the materials.

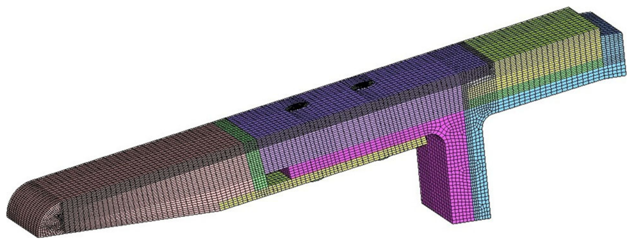

Fig. 12 Finite element model of computational domain

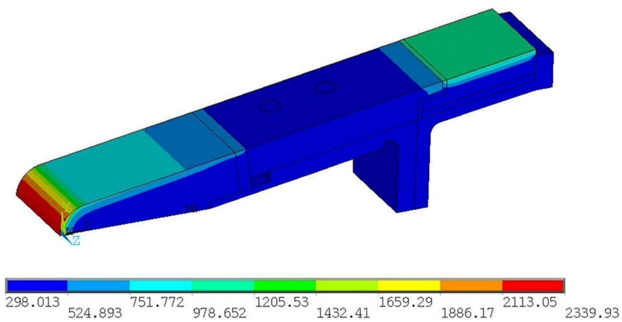

Fig. 13 Temperature contour plot

Fig. 14 Model inside the test chamber

Details of the Second Test

Subsequent to the first test, a repeat test was carried out with the same configuration of the first test with a minor modification in the C–C to molybdenum joint to enable direct contact between C–C and molybdenum bracket. The basic objectives of the test were the same as that of the first test. In addition, the performance of the joint with direct contact of molybdenum bracket with C–C is also included. Since maximum temperature expected on the C–C surface is around 180 °C, it was planned to carry out the test till the surface temperature reaches 180 °C.

Details of the Test Model

The test model used in the first test was refurbished by putting new carbon phenolic leading edge and silica tile. A minor modification is incorporated in the assembly of the C–C specimen to molybdenum plate to ensure its direct contact between them (Fig. 17). The changes made are: The 6 mm dia hole in molybdenum plate is enlarged to 14 mm dia to enable direct butting of molybdenum washer (OD 12 mm) and nut (OD 12 mm) on molybdenum bracket which will simulate the actual joint configuration of C–C nose cap with molybdenum bracket. One CSK M4 fastener is used to connect molybdenum bracket and molybdenum plate which connects with random fibre CP

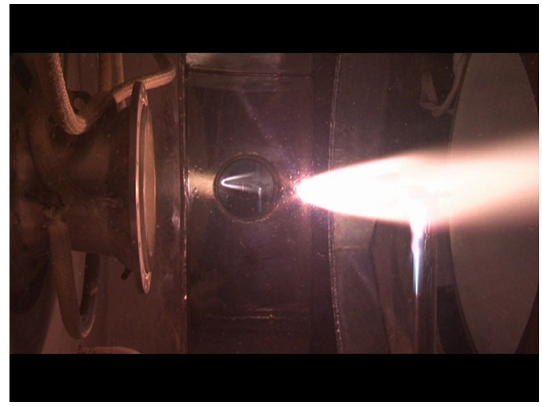

Fig. 15 Testing of the model

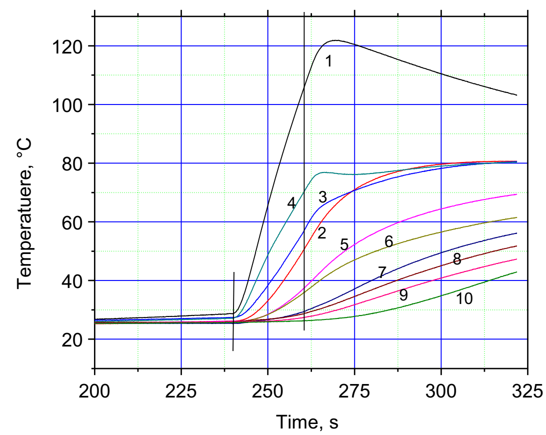

Fig. 16 In-depth temperatures

Instrumentation

In order to evaluate the thermal performance of the joint interface, 10 numbers of K-type thermo couples were provided to measure the C–C back wall temperature, temperature of the molybdenum bolts, interfaces and the brackets. The location of the thermocouples are identical to the first test (Fig. 7) except the thermocouple, Id. no 10, which is now located at a depth of 2 mm from the top surface of the C–C specimen. The NDT was taken after drilling the 2 mm hole to see whether any cracks are

Table 3 Comparison of test results with prediction

Thermocouple ID no.	Predicted temperature, °C (at 22 s)	Measured temperature, °C (at 22 s)
1	84.9	107.0
2	53.6	51.6
3	54.4	58.1
4	52.7	71.4
5	53.7	38.0
6	43.6	36.3
7	37.6	29.6
8	37.6	28.9
9	29.5	27.5
10	30.5	31.1

formed during drilling. Through UT, it was confirmed that there is no crack/delamination present.

Thermal Analysis for the II Test

One of the main objectives of the second test is to expose the model till the surface temperature of C–C reaches 180 °C. Towards this, thermal analysis was carried out to predict the time at which the surface temperature reaches 180 °C in order to arrive at the test duration.

The analysis was carried out for duration of 50 s. The temperature contour plot at the end of 50 s on the C–C surface is shown in Fig. 18. Maximum temperature is found to be 427.9 °C at the gap filler interface. Figure 19 shows the C–C top surface nodal temperatures with time. It may be noted that in the test article, a thermocouple was mounted 2 mm below the top surface of the C–C which is close to the prediction location Tb (Fig. 19). It can be seen that at this location where the thermocouple is mounted, the temperature of 180 °C is reached in 23 s. At the centre of the tile, (Tc), the temperature of 180 °C is reached only at around 38 s. It should also be noted that in flight, the C–C surface experiences the maximum of 180 °C only for duration of 4–5 s.

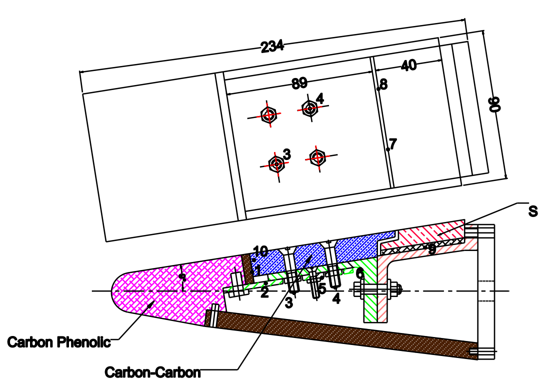

Fig. 17 Configuration for the second test

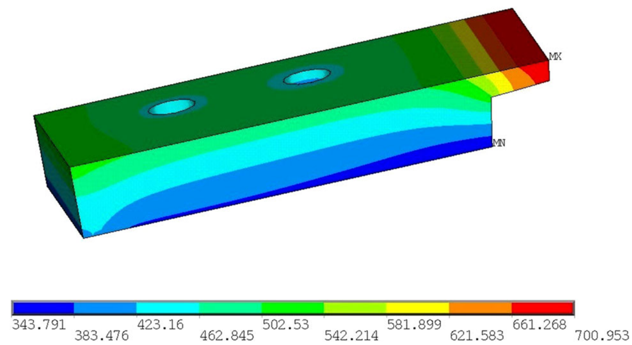

Fig. 18 Temperature contour plot on C–C surface

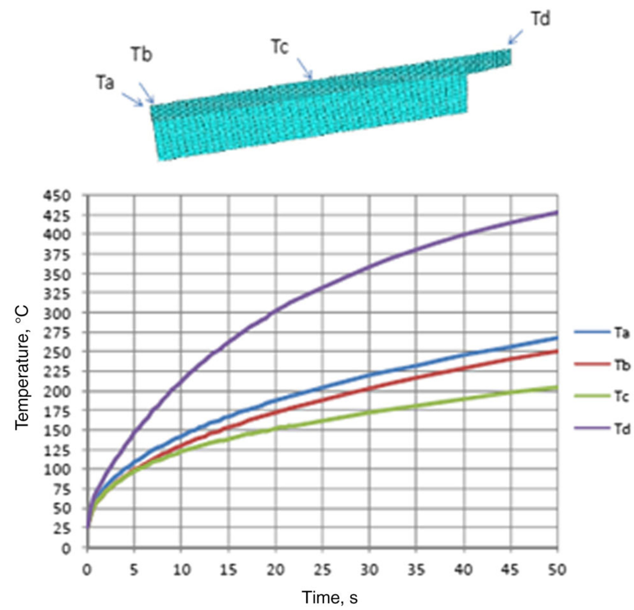

Fig. 19 C–C surface temperature variation with time

Taking into all these considerations, it was decided that the test duration can be fixed in such a way that the test will be stopped when the temperature reading (monitored online during the test) on the C–C surface reaches 180 °C or the test can be continued up to a maximum of 30 s whichever is earlier.

Results of the Second Test

The model was exposed to plasma when the facility was operating at set operating condition corresponds to the calibration test. Online monitoring of the surface temperature of C–C surface was carried out. The temperatures measured were all benign. At the end of 30 s, the temperature of C–C surface reached only 168.8 °C and the test was stopped. Figure 20 shows the measured temperatures

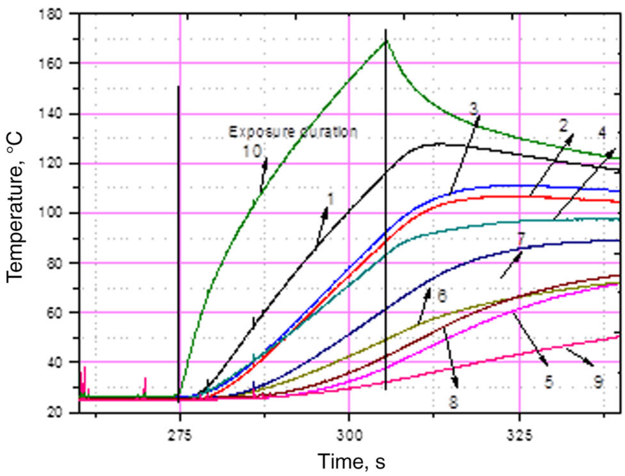

Fig. 20 Temperature measurements

during the test. There was good match between the prediction and the experimental values. After the test, on visual inspection, there were no damages to silica tile or C–C. The UT of carbon–carbon was carried out after the test and no delaminations were observed. No torque relaxation was observed on the molybdenum bolts after the test.

Conclusion

The thermal performance of the C–C with molybdenum interface and C–C to silica tile interface of RLV-TD was evaluated in PWT facility through the simultaneous simulation of heating and shear flow conditions corresponding to a heat flux of 9 W/cm^2. Towards this, two model tests were carried out. The first test was carried out for duration of 22 s to simulate the total heat load expected in the flight. In the second test, model was tested for 30 s and a temperature of 168.8 °C measured at 2 mm depth of C–C.

The visual inspection of the samples after the tests shows no damage or erosion of the C–C or silica tile. The interfaces are intact and no damage is seen. The UT of the C–C carried out after the tests indicate no delaminations or cracks. There was no loosening of torque seen on M6 molybdenum bolts used for fixing C–C with molybdenum brackets. The temperatures measured were all benign.

These tests essentially qualified the joint interface of C–C with molybdenum interface and C–C to silica tile interface of RLV-TD.

References

1. A. Schettino, S. Borrelli, Applicability of SCIROCCO Plasma Wind Tunnel for Testing the Thermal Protection System of FESTIP Concept Vehicles. AIAA-98-1509
2. S.H. Laure, M. Auweter-Kurtz, Experimental Simulation of the Stagnation Point Flow Field of an Aerobraking High Velocity Reentry Vehicle Within a Plasma Wind Tunnel. AIAA 94-2569
3. M. Auweter-Kurtz, H.L. Kurtz, S. Lauret, Plasma generators for re-entry simulation. J. Propuls. Power **12**, 1053–1061 (1996)
4. A. Pillai, et al., Thermal Performance Evaluation of Carbon Phenolic Tiles in the Fore Body Region of HSP Crew Module Using Plasma Wind Tunnel Facility. IHMTC 2015-1412
5. M. Ajith, A. Pillai, et al., Non-equilibrium Flow Simulation of Plasma Wind Tunnel Test of HSP Crew Module. IHMTC 2015-595